C. R. Mecanique 330 (2002) 709–716

Adaptive remeshing for ductile fracture prediction in metal forming

Houman Borouchaki [a], Abdelhakim Cherouat [a], Patrick Laug [b], Khemais Saanouni [a]

[a] **Université de technologie de Troyes, GSM-LASMIS, BP 2060, 10010 Troyes cedex, France**

[b] **INRIA-Rocquencourt, BP 105, 78153 Le Chesnay cedex, France**

Received 25 June 2002; accepted 1 July 2002

Note presented by Évariste Sanchez-Palencia.

Abstract The analysis of mechanical structures using the Finite Element Method in the framework of large elastoplastic strain, needs frequent remeshing of the deformed domain during computation. Indeed, the remeshing is due to the large geometrical distortion of finite elements and the adaptation to the physical behavior of the solution. This paper gives the necessary steps to remesh a mechanical structure during large elastoplastic deformations with damage. An important part of this process is constituted by geometrical and physical error estimates. The proposed method is integrated in a computational environment using the ABAQUS/Explicit solver and the BL2D-V2 adaptive mesher. ***To cite this article: H. Borouchaki et al., C. R. Mecanique 330 (2002) 709–716.***

solids and structures / moving mesh / adaptive meshing / remeshing in large deformation / error estimate / finite plastic strain / ductile damage / orthogonal cutting / crushing

Remaillage adaptatif pour la prévision de la rupture ductile en mise en forme

Résumé Le calcul des structures en grandes déformations élastoplastiques par la méthode des éléments finis nécessite de recourir à des remaillages fréquents du domaine de calcul au cours de la résolution. En effet, la fiabilité et la performance d'un tel calcul dépendent fortement de l'évolution géométrique et physique du domaine de résolution. Cet article présente les différentes étapes nécessaires au remaillage d'une structure mécanique au cours de grandes deformations plastiques avec endommagement. Une part importante de ce processus est constituée par les estimations d'erreur à caractère géométrique et physique. La méthode proposée a été intégrée dans un environnement de calcul utilisant le solveur ABAQUS/Explicit et le mailleur adaptatif BL2D-V2. ***Pour citer cet article : H. Borouchaki et al., C. R. Mecanique 330 (2002) 709–716.***

solides et structures / maillage mobile / maillage adaptatif / remaillage / estimateur d'erreur / grandes deformations plastiques / endommagement ductile / découpage orthogonal / écrasement

E-mail addresses: Houman.Borouchaki@utt.fr (H. Borouchaki); Abdelhakim.Cherouat@utt.fr (A. Cherouat); Patrick.Laug@inria.fr (P. Laug); Khemais.Saanouni@utt.fr (K. Saanouni).

S1631-0721(02)01519-X/FLA

Version française abrégée

La mécanique non linéaire des solides traite de la modélisation et du calcul des structures à fortes non-linéarités géométriques (transformations finies, contact unilatéral évolutif, etc.) et matérielles (plasticité, viscoplasticité, écrouissage, endommagement, etc.). Il s'agit de simuler numériquement le comportement d'une structure mécanique soumise à diverses sollicitations thermomécaniques plus ou moins complexes afin d'améliorer sa tenue (durée de vie), voire d'optimiser le procédé même de fabrication de l'objet. La fiabilité et la performance d'une telle simulation repose sur celles des outils théoriques (relations constitutives représentant les phénomènes physiques), numériques (algorithmes d'intégration des EDO, schémas de résolution des systèmes non linéaires, etc.) et géométriques (représentation géométrique de l'objet, discrétisation en éléments finis ou maillage, remaillage et maillage adaptatif en cours de la simulation).

Les aspects théoriques (modèles de plasticité avec endommagement [1,2]) et numériques (algorithmiques) ont été largement développés depuis fort longtemps, et plusieurs outils plus ou moins performants ont été proposés [4]. Pour les aspects géométriques, la représentation 2D ou 3D d'objets ainsi que leur discrétisation initiale par éléments finis ont fait également l'objet de grands efforts de développement (voir [3] pour une synthèse). Quant aux remaillages et aux maillages adaptatifs, nécessaires aux problèmes fortement non linéaires, ils sont en plein développement (cf. [5,6,4] entre autres). Mentionnons que la principale difficulté réside dans le fait qu'en grandes deformations la géométrie du domaine est variable et ne peut être définie d'une manière explicite.

Soit $\Omega \in \mathbb{R}^2$ un domaine défini à partir de sa frontière Γ. On suppose que cette dernière est donnée sous une forme discrète constituée d'un ensemble de segments droits qui constitue une discrétisation initiale $\mathcal{T}(\Gamma)$ de Γ. De plus, on suppose que la déformation finale est obtenue itérativement par pas de « petites » déformations. Le remaillage est appliqué après chaque incrément de déformation selon le schéma suivant :

- définition de la nouvelle géométrie $\mathcal{G}(\Gamma)$ après déformation,
- estimation d'erreur géométrique (écart entre la nouvelle géométrie $\mathcal{G}(\Gamma)$ due aux divers types de déformation et la discrétisation courante $\mathcal{T}(\Gamma)$ de la frontière) ; définition d'une carte de taille géométrique $\mathcal{H}_g(\Gamma)$,
- estimation d'erreur physique (écart entre la solution physique $\mathcal{S}(\Omega)$ obtenue sur Ω et une solution idéale « lisse » considérée comme la solution de référence) ; définition d'une première carte de taille physique $\mathcal{H}_{\varphi_1}(\Omega)$,
- adaptation de la taille des éléments du maillage en fonction de l'endommagement ; définition d'une deuxième carte de taille physique $\mathcal{H}_{\varphi_2}(\Omega)$,
- intersection des cartes de taille $\mathcal{H}_g(\Gamma)$, $\mathcal{H}_{\varphi_1}(\Omega)$ et $\mathcal{H}_{\varphi_2}(\Omega)$; définition d'une carte de taille unique $\mathcal{H}(\Omega)$,
- rediscrétisation adaptative $\widetilde{\mathcal{T}}(\Gamma)$ de la frontière du domaine par rapport à $\mathcal{H}(\Omega)$,
- remaillage adaptatif $\widetilde{\mathcal{T}}(\Omega)$ du domaine par rapport à $\mathcal{H}(\Omega)$.

1. Introduction

The object of non-linear solid and structural mechanics is the modeling and the computation of structures with strong non-linearities, both geometric (finite transformations, evolving unilateral contact, etc.) and physical (plasticity, hardening, damage, etc.). The aim is to simulate numerically the behavior of a mechanical object subjected to various mechanical loadings, in order to improve its endurance or even to optimize its manufacturing process. The reliability and the performance of such a simulation are based on different types of tools: theoretical (constitutive relations representing the physical phenomena), numerical (algorithms to integrate the ODEs, schemes to solve non-linear systems, etc.) and geometric (representation of the object shape, finite element discretization or meshing, remeshing and adaptive meshing during the simulation).

Theoretical aspects (plasticity with damage models) and numerical aspects have been widely developed for many years, and more or less performing tools have been proposed (see [1,2] for more details). Concerning geometric aspects, the 2D or 3D object representation, as well as the initial finite element discretization, have also given rise to many development efforts (see [3] for a synthesis). As for adaptive remeshing, which is necessary for strongly non-linear problems, the interest is really high nowadays and the proposed solutions are not quite satisfactory (see [4–6] amongst others). Let us mention that the main difficulty lies in the fact that, in large deformations, the domain geometry is variable and cannot be defined in an explicit way.

In this article, we are interested in the problem of remeshing a mechanical structure subjected to large plastic deformations with damage, using a local formulation. A general scheme, constituted by several steps necessary to an almost optimal representation of the evolving domain, is presented. These steps are divided into two main categories: the definition of the boundary of the deformed domain and the adaptive remeshing of the domain. The remeshing is governed by a mesh element size map representing the satisfaction of the underlying geometry of the deformed domain, the improvement of the accuracy of the desired physical solution and the convergence of the mechanical process as well. It must be applied at each iteration of small deformation increment, which must be sufficiently small with respect to the specified minimal size of mesh elements.

2. General remeshing scheme

We consider a part modeled by a computational domain Ω of $\mathbb{R}^2$ defined from its boundary Γ, which is supposed to be given in a discrete form as a set of straight segments. This set constitutes an initial discretization $\mathcal{T}(\Gamma)$ of the boundary Γ. The analytical definition of this boundary is used for obtaining the initial discrete geometry only, but not for remeshing because of the large variations of the domain geometry during computation. Moreover, the final deformation is assumed to be obtained iteratively by 'small' deformations (which is the case in the framework of an explicit integration scheme to solve the problem). Remeshing is applied after each deformation increment, using the following scheme:

- definition of the new geometry $\mathcal{G}(\Gamma)$ after deformation,
- geometric error estimation (gap between the new geometry $\mathcal{G}(\Gamma)$ and the current boundary discretization $\mathcal{T}(\Gamma)$); definition of a geometric size map $\mathcal{H}_g(\Gamma)$ necessary to rediscretize the boundary Γ,
- physical error estimation (gap between the physical solution $\mathcal{S}(\Omega)$ obtained in Ω and an ideal 'smooth' solution considered as the reference solution); definition of a first physical size map $\mathcal{H}_{\varphi_1}(\Omega)$ necessary to govern the remeshing of domain Ω,
- adaptation of the mesh element size with respect to the damage; definition of a second physical size map $\mathcal{H}_{\varphi_2}(\Omega)$ also necessary to govern the remeshing of domain Ω,
- intersection of the size maps $\mathcal{H}_g(\Gamma)$, $\mathcal{H}_{\varphi_1}(\Omega)$ and $\mathcal{H}_{\varphi_2}(\Omega)$; definition of a unique size map $\mathcal{H}(\Omega)$,
- adaptive rediscretization $\widetilde{\mathcal{T}}(\Gamma)$ of the domain boundary with respect to $\mathcal{H}(\Omega)$,
- adaptive remeshing $\widetilde{\mathcal{T}}(\Omega)$ of the domain with respect to $\mathcal{H}(\Omega)$.

2.1. Definition of $\mathcal{G}(\Gamma)$

The new geometry $\mathcal{G}(\Gamma)$ of Γ is determined by three types of deformation:

- *free deformations:* this type concerns the deformations due to mechanical constraints (for instance equilibrium conditions), freely in the plane. In this case, the new geometry of the piece after deformation is only defined by the new position of the boundary nodes as well as their connections;
- *bounded deformations:* these are the deformations limited by a contact with a second domain whose geometry is fixed (the tool is assumed to be rigid). In this case, the piece takes the geometric shape of the tool and thus its geometry after deformation is that of the tool;
- *imposed deformations:* the third type of deformation concerns deformations obtained by elimination of totally damaged elements. In this case, a new geometry of different topology can possibly be obtained.

2.2. Definition of size maps $\mathcal{H}_g(\Gamma)$, $\mathcal{H}_{\varphi_1}(\Omega)$ et $\mathcal{H}_{\varphi_2}(\Omega)$

To estimate the size at any boundary node, several factors must be considered. If the node is free, the size is proportional to the curvature radius of the new piece boundary. If it is bounded, the size is proportional to the curvature radius of the neighboring part of the tool. Finally, if the node is imposed, the size depends on the edge lengths of the current boundary discretization that are adjacent to the node. All these size specifications define the size $\mathcal{H}_g(\Gamma)$. To quantify the gap between the physical solution obtained by computation and the exact solution, the interpolation error can be majored depending on the mesh element sizes [7]. Formally, a size map $\mathcal{H}_{\varphi_1}(\Omega)$ associated to the nodes of $\mathcal{T}(\Omega)$ is defined. In the model of applied mechanics, macroscopic cracks are deduced from the suppression of totally damaged elements. To approach the physical reality at best, a minimal size is imposed to the damaged elements. This leads to defining another size map $\mathcal{H}_{\varphi_2}(\Omega)$ associated with $\mathcal{T}(\Omega)$.

2.3. Generation of the discretization $\widetilde{\mathcal{T}}(\Gamma)$ and the mesh $\widetilde{\mathcal{T}}(\Omega)$

From the three size maps $\mathcal{H}_g(\Gamma)$, $\mathcal{H}_{\varphi_1}(\Omega)$ and $\mathcal{H}_{\varphi_2}(\Omega)$, a size map $\mathcal{H}(\Omega)$, intersection of the first three maps, is defined (which consists to consider the smallest size in $\mathcal{H}_g(\Gamma)$, $\mathcal{H}_{\varphi_1}(\Gamma)$ and $\mathcal{H}_{\varphi_2}(\Gamma)$, where $\mathcal{H}_{\varphi_i}(\Gamma)$ is the restriction of the size $\mathcal{H}_{\varphi_i}(\Omega)$ to the boundary Γ of Ω, $i = 1, 2$). This new map is used to govern the adapted remeshing of the piece which includes two phases, the boundary rediscretization and the remeshing of the piece based on this discretization. The boundary rediscretization consists to generate a new discretization of Γ conforming to the size map $\mathcal{H}(\Gamma)$. The piece is then entirely remeshed, from the boundary rediscretization, using a classical method of h-adaptation based on a combined advancing-front – Delaunay approach in order to be conform to the map $\mathcal{H}(\Omega)$.

3. Definition of the boundary of the deformed domain

To define the new deformed boundary, two successive steps are executed: the identification of the imposed nodes, then of the bounded nodes. In each case, the underlying geometry is defined using a different interpolation scheme to ensure the corresponding geometric continuity.

3.1. Identification of the imposed nodes

At first, all the damaged elements are eliminated from the piece. This may lead to a new boundary having multiple points or several connected components. In the first case, the elements sharing multiple nodes are suppressed, and this procedure is repeated if new multiple points appear. In the second case, although the choice of the connected components of the piece that must remain for the rest of the computation can only be arbitrary, a heuristic consists to consider the largest one.

3.2. Identification of the bounded nodes

The idea consists to a priori identify the nodes of the piece which are in contact with the tool (if the displacement step is applied to the tool) by considering Hausdorff distance from the piece to the tool. In 2D, this consists of associating, to each edge of the discretization of the piece and the tool, a region centered in this edge, and of examining the possible intersections between the regions associated to the piece and those associated to the tool. A node of the piece is then classified as bounded if it belongs to one of the regions associated with the tool, or if a node of the tool belongs to one of the regions associated with the piece and including this node. Formally, the region $\mathcal{R}_\delta(e)$ associated with the edge e is defined by:

$$\mathcal{R}_\delta(e) = \left\{ X \in \mathbb{R}^2,\ d(X, e) \leqslant \delta \right\}$$

where $d(X, e)$ is the distance from X to e, and δ the displacement step of the tool. In the case where the orthogonal projection of X on the straight line including edge e is inside e, then the distance $d(X, e)$ is equal to the distance from X to its projected point and, in the opposite case, $d(X, e)$ is equal to the smallest distance from X to the extremities of e.

3.3. Interpolation schemes

The geometry (after deformation) of the part can be defined in two ways, either by preserving a geometry close to the one before deformation, or by defining a new 'smoother' geometry. In the first case, the new geometry is simply defined by the current discretization of the boundary, and the new mesh nodes of the piece are placed on the elements of this discretization. In the second case, the new geometry is defined in two dimensions by a smooth curve interpolating the nodes and/or other geometric features of the current boundary discretization. The new nodes are then placed on this curve. The advantage of this second approach (which seems heavier) is that the geometry of the piece remains smooth during its deformation.

In this second approach, an interpolating scheme with cubic splines, globally of C^2 continuity, according to a technique proposed by de Boor, is used. This comes to solving a linear system whose equations

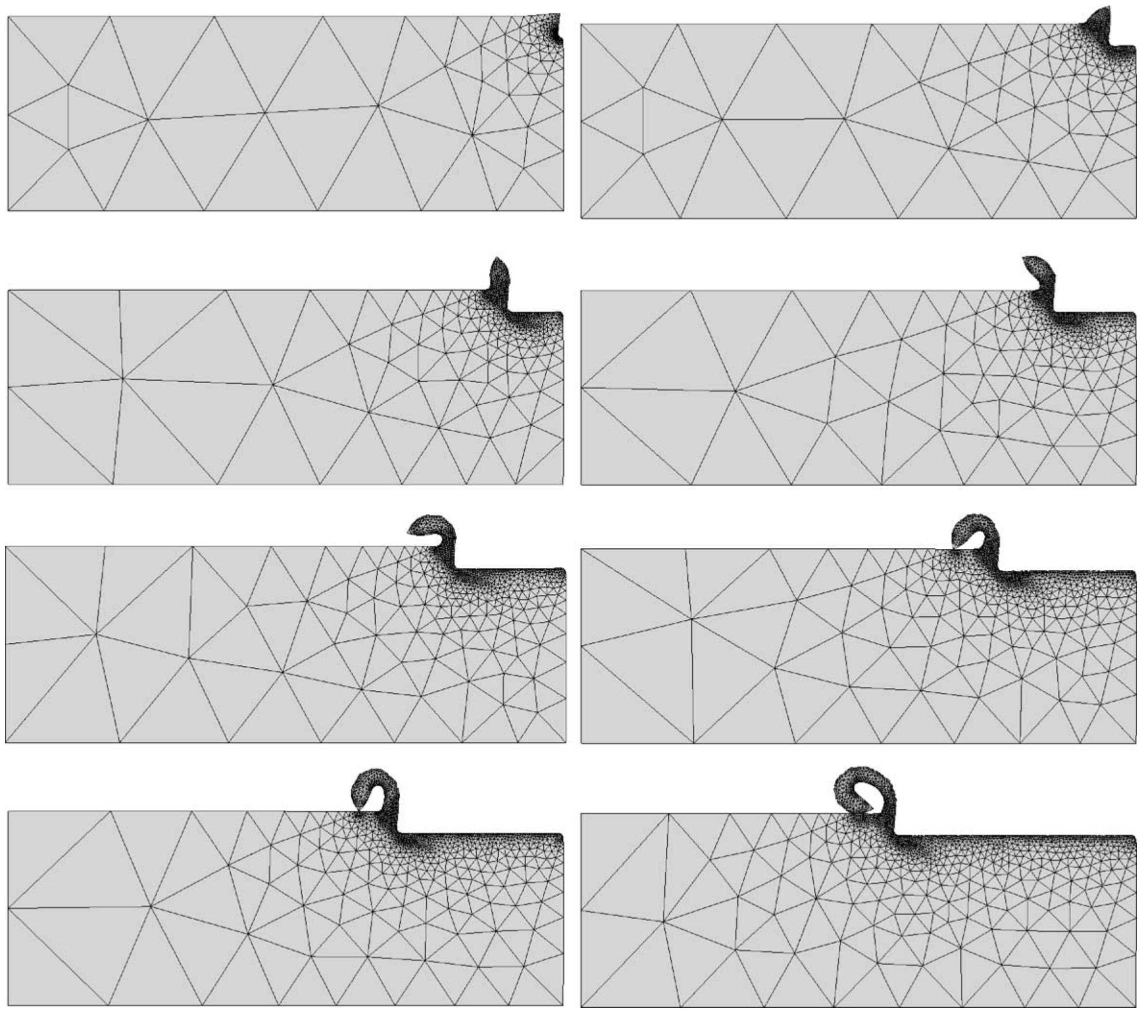

Figure 1. Cutting at increments 10, 50, 100, 150, 200, 250, 300 and 436 and ship formation.

Figure 1. *Découpage aux incréments* 10, 50, 100, 150, 200, 250, 300 *et* 436 *et formation du copeau.*

represent the continuities of order 0, 1 and 2 at the extremities of each cubic piece. This approach is only applied to the free or bounded nodes. Indeed, the boundary obtained by element removal does not exactly reflect the physical reality, and an interpolation of order 2 cannot be applied. On the other hand, a non interpolating smoothing can produce adequate results.

4. Application examples

In this section, two examples are presented, namely the orthogonal cutting by chip formation, and the sidepressing of an infinite cylinder. In each case, the 2D structure is meshed with three-node triangular linear elements (assuming plane strain hypothesis). The material is supposed isotropic elastoplastic, isothermal with non linear isotropic hardening and isotropic ductile damage. Its mechanical characteristics are those of an aluminum with $E = 85000$ MPa, $\nu = 0.38$, $\sigma_y = 120$ MPa, $Q = 600$ MPa, $b = 3$, $S = 0.1$, $s = 1$, $\beta = 1$ and $a = c = 0$ (no kinematical hardening). The theoretical and numerical aspects of this fully coupled

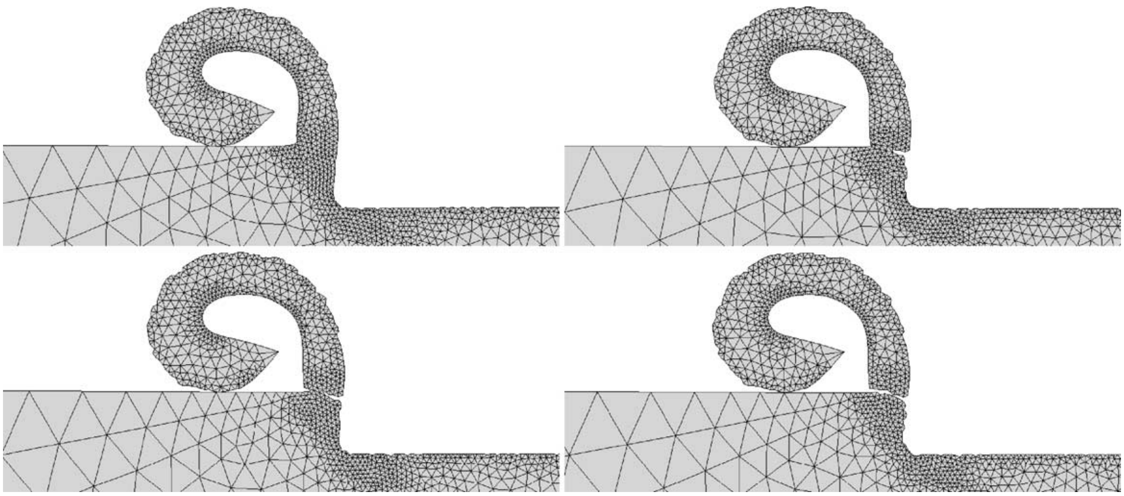

Figure 2. Cutting at increments 480, 493, 495 and 497 and ship breaking.

Figure 2. *Découpage aux incréments* 480, 493, 495 *et* 497 *et rupture du copeau.*

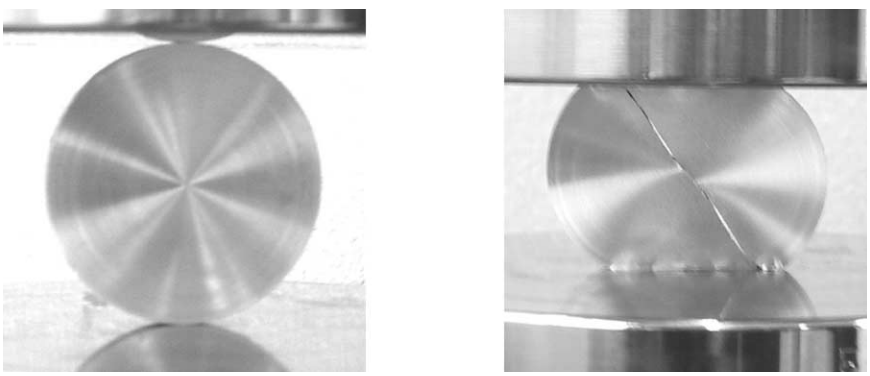

Figure 3. Experimentation of the sidepressing process.

Figure 3. *Expérimentation du procédé d'écrasement.*

contitutive equations can be found in [8]. In these examples, the ABAQUS/Explicit solver and the BL2D-V2 adaptive mesher [9] were used.

4.1. Orthogonal cutting

Manufacturing by chips removing is one of the most frequently used operation for making pieces of structures. By modeling the material cutting, it is possible to determine the optimal cutting conditions and the thermo-mechanical characteristics of the manufactured piece. The example considered concerns the simulation of a chip formation when massive parts are manufactured by 2D orthogonal cutting. The tool is assumed undeformable and its edge has an orthogonal cutting angle of 5° and is assimilated to a circle portion with a very small radius ($R = 0.2$ mm). The cutting speed is fixed to 0.1 mm/s and the removed layer is 2 mm thick. Fig. 1 shows meshes adapted to the damaged fields of the piece corresponding to different displacements of the tool. It can be noticed on these pictures that a chip forms owing to the deformation imposed by elements removal. Besides, the areas in contact with the tool (corresponding to the

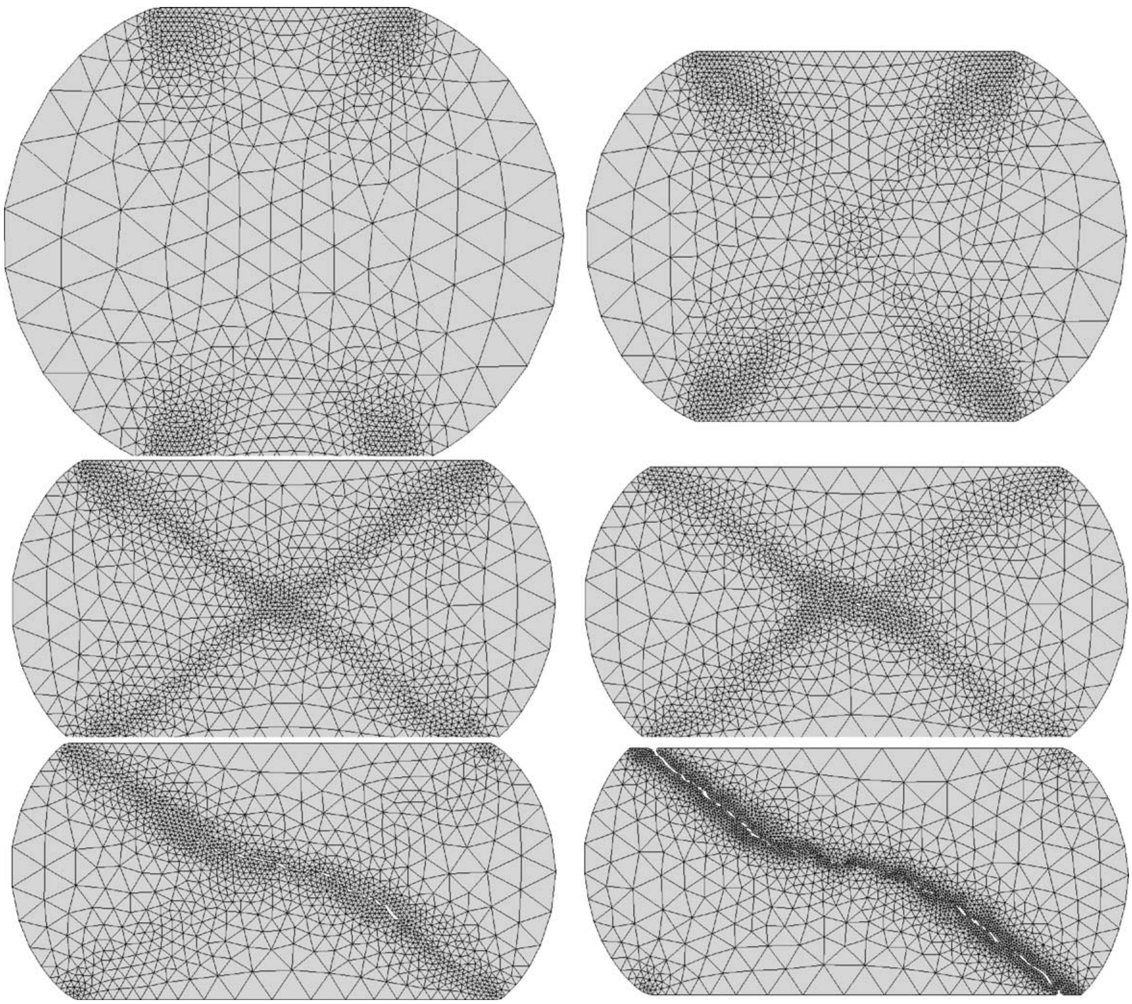

Figure 4. Sidepressing at increments 30, 50, 75, 79, 86 and 92.

Figure 4. *Écrasement aux incréments* 30, 50, 75, 79, 86 *et* 92.

bounded deformation), as well as the damaged areas (corresponding to the control of physical error) are refined. Fig. 2 shows the beginning, the propagation and the segmentation of the previous chip due to the tool progression. As this figure and the experimentation [10,11] show, the breaking is located at the base of the chip.

4.2. Sidepressing of a cylinder

This process is generally used for ductile materials to control their cold forgeability conditions. In this example, a cylindrical piece of diameter 50 mm is pressed between two planar tools, assumed infinitely long, which tend to flatten it. The assumed rigid tools are composed of both the mobile upper plane through which the imposed displacements are exerted (0.1 mm by increment) and the fixed lower plane. Fig. 4 shows the meshes adapted to the damage fields of the cylinder corresponding to different displacements of the mobile tool. First, the cylinder damage is located on two diagonal shear bands according to the well-known ‘blacksmith cross’, frequently seen during experimentation [12]. Then, the beginning of the macroscopic crack occurs at the intersection of the two bands (center of the cylinder) for a height reduction of 40%. This crack propagates along one of the bands until the total breaking which splits the cylinder in two parts. Fig. 3 shows an experimentation of this process and confirms the result obtained by simulation.

5. Conclusions and future prospects

A solving scheme based on a technique of adaptive remeshing for problems in large elastoplastic deformations, taking into account the damage, has been proposed. The numerical solving of several types of problems has validated the proposed approach and proved its efficiency. However, several points can be improved. In particular, smoothing the parts of the piece boundary obtained by imposed deformations would reduce significantly the number of mesh elements. Moreover, an a priori identification of contacts following a geometric approach would reduce the problem resolution time. Also, the problems met being often of tridimensional nature, the generalization of this solving scheme in three dimensions seems the most important point to study.

References

[1] J. Lemaitre, A Course on Damage Mechanics, Springer-Verlag, 1992.
[2] K. Saanouni (Ed.), Numerical Modelling in Damage Mechanics, NUMEDAM ’00, Revue européenne des éléments finis 10 (2–4) (2001).
[3] P.J. Frey, P.L. George, Maillage. Applications aux éléments finis, Hermès, Paris, 1999.
[4] L. Fourment, Estimation d’erreurs et maillage adaptatif pour la simulation numérique de procédés de mise en forme, Thèse de l’école des Mines de Paris, CEMEF, Sophia-Antipolis, 1992.
[5] M. Homsi, L. Morançay, J.M. Roelandt, Techniques de remaillage appliquées au découpage des métaux, Revue européenne des éléments finis 5 (3) (1996) 297–321.
[6] P. Coorevits, J.P. Dumeau, J.P. Pelle, Analyses éléments finis adaptatives pour les structures tridimensionnelles en élasticité, Revue européenne des éléments finis 5 (3) (1996) 341–373.
[7] H. Borouchaki, D. Chapelle, P.L. George, P. Laug, P.J. Frey, Estimateurs d’erreur géométriques et adaptation de maillage, Chapitre 9 in Maillage et adaptation (sous la direction de P.L. George), in: Mécanique et Ingénierie des Matériaux, méthodes numériques, Hermès, 2001, pp. 279–310.
[8] K. Saanouni, A. Cherouat, Y. Hammi, Numerical aspects of finite elastoplasticity with isotropic ductile damage for metal forming, Revue européenne des éléments finis 10 (2–4) (2001) 327–351.
[9] P. Laug, H. Borouchaki, BL2D-V2 : mailleur adaptatif bidimensionnel de domaine à frontière variable, Description technique, INRIA, 2001.
[10] E. Ceretti, M. Lucchi, T. Altan, FEM simulation of orthogonal cutting: serrated chip formation, J. Materials Processing Technology 95 (1999) 17–26.
[11] S.L. Ko, D.A. Dornfeld, Burr formation and fracture in oblique cutting, J. Materials Processing Technology 62 (1996) 24–36.
[12] E.D. Mielnick, Metalworking Science and Engineering, McGraw-Hill, 1999.